Design Evolution of a Typical Aircraft Engine Mount Bracket Using FE Based Optimisation Technique

Revanasidda Benur[1] and Akshatha H T[2]

[1]former project trainee, [2] Senior Scientist, Centre for Civil Aircraft Design and Development, CSIR-National Aerospace Laboratories, Bangalore, INDIA

Abstract: In the present scenario of aerospace sector, structure weight is considered to be major factor in deciding the performance, efficiency etc. One of the potential major weight reduction contributor is the structural airframe. Structure optimization which deals with the structural weight reduction without compromising strength is a complicated, time consuming and experience based process.In this paper, optimisation of a critical bracket of flying prototype aircraft engine mount is demonstrated using a Finite Element(FE) based optimisation software Altairoptistruct. The optimisation is carried in two steps. Topology optimisation, which is anelastic energy based optimisation to search for anoptimum material distribution followed by a size optimisationto get a final solution of the product. The optimisation was carried with an objective of weight reduction satisfying the strength requirements. As an outcome of this exercise, the final product ready for realisation is lighter in weight by around 20% compared to the initial design.

Keywords: topology optimisation, size optimisation, engine mount bracket, weight reduction.

I. Introduction:

In aircraft, the weight saving in anyform is the most sought after parameter at any stage of the realisation of aircraft as it would greatly help in improvisation of its performance, increase in payload,etc. The present work is based on the weight optimisation carried on an engine mount bracket. The engine mount brackets are most critical components of the aircraft as their failure may lead to catastrophic failure of the system and hence the aircraft. Apart from safety issue, the main design driver for these brackets is the influence of multiple load cases considered as per the aviation regulations. One of the critical load case is propeller malfunction case where the load resulting would be of the multiple order of load during engine operating condition. Hence any design change would be based on the multiple load paths arising from different loading condition.

The bracket considered in this work is a part of fail-safe engine mount assembly. The main function of the bracket is to transfer the loads from the engine connecting rods to the frames. The possibility of weight reduction in these brackets was explored subjected to strength as a constrain(von misesstress).As a starter, a static global analysis is carried to extract the local loads at desirable location. Due to well defined local load path,optimisation of an isolated bracket would be carried out initially which could be extended to the global analysis later. The basic design of the bracket considered is a combination of top aluminium alloy bracket connected with a bottom steel bracket through bolts. Hence the optimisation was carried out in steps considering the bracketindividually to arrive at the final design solution.

II. Literature review.

Hervandil M. Sant'Anna and Jun S. O. Fonseca [1] presented the problem of volume minimization of two-dimensional continuous structures with compliance and stress constraints. Here they conducted topology optimisation to check for optimal material distribution, where they cut down geometry into number of simple pieces, approximating the displacement methods. A first neighbourhood filter was implemented to minimize the effects of checkerboard patterns and mesh dependency, two common problems associated to topology optimization. Von-misesstress constrains was used for arriving at the final solution. As stress constrained problems have a difficulty of stress singularity, the feasible domain is modified using a mathematical perturbation technique, the epsilon-relaxation. From the exercise, they concluded that problems considering stress constraints require a more refined finite element mesh to obtain better solutions for engineering problems.

Lars Krog et al [2], in their paper, effectively demonstrated the application of energy based topology optimization methods for design of aircraft wing box ribs for airbus project. The work focused on use of both FE based global and local analysis approaches as the ribs are embedded inside the aircraft wing and subjected multiple loads like fuel pressure, structural loads etc. Here, the analysts considered alternate optimization formulation approaches like min-max formulation, energy measure based load case for achieving minimum weight of the component.

In the work by Lee et al[3], the authors tried to solve the structural topology optimisation problem with stress constrain in place of regular compliance minimisation in case of deign dependent loading. They compared the results from both methods for the same geometry and loading. It was concluded that topology obtained from these methods are vastly different and the sizing optimization of a compliance solution may not lead to an optimum.

III. Interactive Methodology

The component is optimised in a standard 2 step FE based optimization process. In the first step a search for optimum material distribution using minimum compliance formulation within the designable space is run. The next step involves the finite element sizing and shape optimization to satisfy strength based parameters. Here, the user has an ample freedom to redefine and reanalyse the design as per requirement. This interactive process flow is depicted in the following diagram(fig 1).

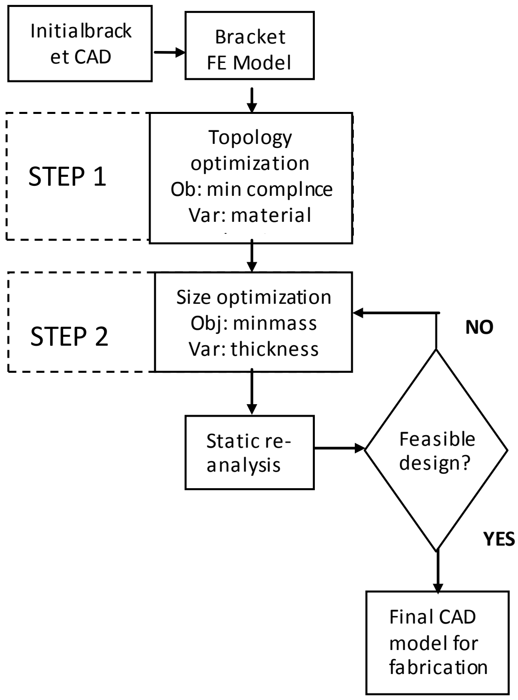


Fig 1 Flow diagram for a 2 step optimisation problem

Topology optimizationin continuum structure is a strain energy based formulation,which is the most efficient way to avoid unnecessary material present in the desired designable space satisfying the material distribution based on the load path. This step does not provide the component sizing as none of the structural design parameters such as displacement, stress or buckling are considered while analysis. Initially a designable space "box like model" regardless of actual dimensions is built. The base box is associated with an artificial material and loaded with critical structural load as shown in the figure 2.

a) b)

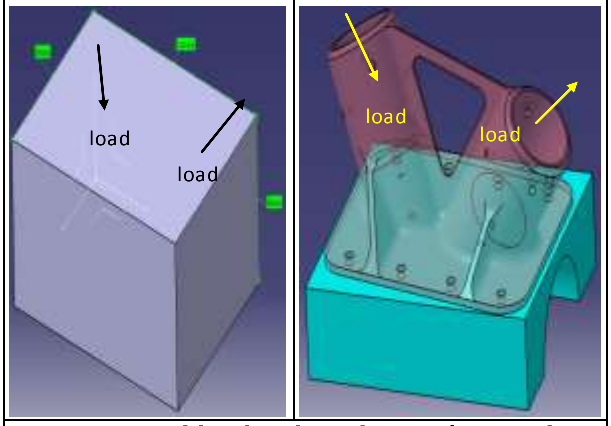


Fig 2: Box like baseline design for topology optimisation of a) top aluminium bracket b) bottom steel bracket

In the topology problem, the artificial material is associatedto each finite element with the variable material density ρ and variable stiffness $K(\rho,q)=\rho^q K$ where K is the real stiffness of the material and q the penalisation factor for the element with intermediate density in the design space. During topology optimisation process, the designable area is calculated for each finite element to associate the material density to be either 0,if the element may not lay in the load path or 1 as the case may be load diffusing path. By this simple formulation, the total strain energy measure *U* can be taken as the objective function and material distribution can be represented as[2],

Minimize$U$ $n=1,\ldots N$

Subject to$\Sigma^N_{n=1}\rho_n V_n=V_0$

$0<\rho_n<1$ $n=1,\ldots N$

Where ,ρ_n=material density of n^{th} finite element, V_n= volume of n^{th} finite element, V_0=volume of material for topology optimisation, N= total number of finite elements.

This represents the simplified scientific version of the classical elastic energy based minimum global compliance topology optimization problem which can be expressed in other way as maximization of stiffness for a given load case.

Once the basic material distribution based on the stiffness requirement is established, the detailing is carried out using size optimization. Here, the technique involves the declaration of finite element property as a function of design variable like thickness, cross section,spring stiffness etc. Then the property,p to be optimisedcan be expressed in terms of[3]

$$P=C_0+\Sigma DV_i.C_i$$

Where C_0 and C_i are the linear factor associated with the design variable DV.For the present problem, the design-variable-to-property relationship turns into,

$$T=DV_i$$

Where T is the gauge thickness associated with the property. The constraint to be satisfied is the yield stress of the respective material.

IV. Results and discussion:

The first step of topology optimisationyield the basic material density flow for the given load. The result at an intermediate stage during optimisation for top aluminum and bottom steel brackets is shown in figure 3. As the load(rod) for top bracket was of axial in nature the baseline design is clearly indicating the clustering of material in an inclined cylindrical nature. The applied loads being opposite in nature, itislikely to create twisting moment effect on the bracket. Hence it was required to bridge the cylinders with an intermediate vertical plate. Since top bracket is attached to the bottom bracket through series of bolts, the bottom bracket design is guided by the predefined attachment points. The initial topology optimisation resulted in a truss like structure; the additional stiffening is introduced in the form of gussets at the end of cylinders.

(a)	(b)

Vol. 1, Issue. 3, June 2015 ISSN (Online): 2394-8868

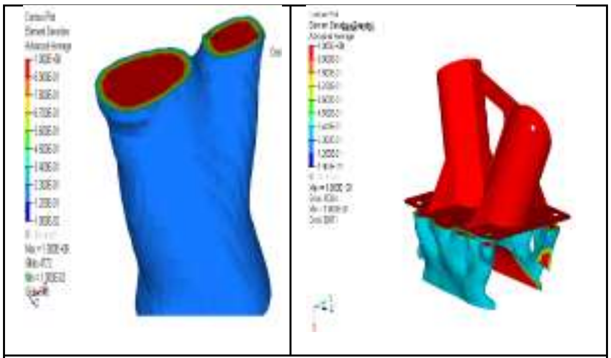


Fig 3. Density flow at intermediate stage of topology optimisation for
a) top aluminium bracket b) bottom steel bracket

a) b)

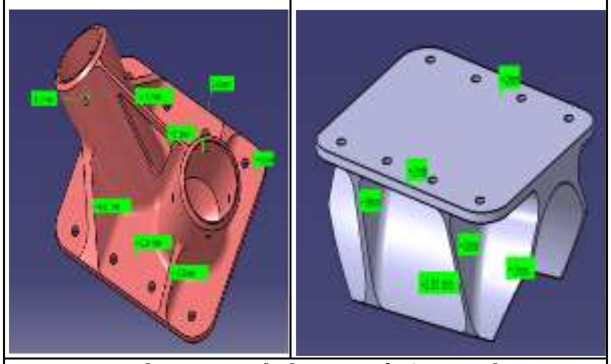
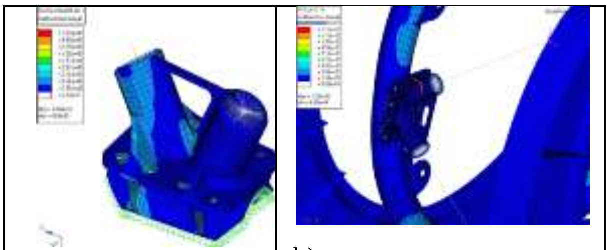


Fig 4. Final optimised design of a) top aluminium bracket b) bottom steel bracket

Based on the design layout from minimisation of inverse of stiffness (compliance) analysis, a fabrication standard design is modelled and the second step of minimisation of weight of the component with variable thickness and von mises stress constrain is performed.

The thickness bounds were varied and iterative analysis was carried out to search for the best



a) b)

Fig 5. Stress analysis of the final product in a) isolated domain b) assembly domain

possible design satisfying the strength constraint. Based on the result of this min-min 2 step optimisation formulation, considering the manufacturing and assembly constraints, the design was finalised with sizing as shown in fig 4. To cross check the feasibility of the final product, the final static analysis is carried out in aisolated local(fig 5a) and assembled global(fig 5b)configurations.The new weight reduced design was found to satisfy the strength and stiffness of the bracket and the counter attachments.

V. Conclusion:

In this work, the design evolution of a critical aircraft bracket subjected to various loads using a finite element based optimisation technique is successfully demonstrated on a real time project.Here, the design process was interactive way of achieving the desired end product where the parameters could be modified at any intermediate stage. The overall weight saving from this technique was 20.17% for top aluminium bracket and 17.3% for the bottom steel bracket as compared to the initial design. This is considered to be a very good weight saving by the aircraft weight optimisation standard. The bracket design sensitivity through iterative analysis and a final combined global analysis is an evidence of effective use of software based optimisation technique for industry application where the real time projects require time and quality based solution to increase the performance and productivity. The authors intend to further simply the process for a multiple load scenario and other potential constrains such as attachment loads and displacement. In future, it is highly likely to adopt the same methodology for other bracket design for the same aircraft components.

VI. References:

1) Hervandil M. Sant'Anna and Jun S. O. Fonseca, "Topology Optimization of Continuum Two-Dimensional Structures under Compliance and Stress Constraints" Mechanicacomputacional, vol XXI, Argentina, October 2002.

2) Lars Krog, Alastair Tucker and Matin Kemp and Richard Boyd, “Topology optimisation of aircraft wing box ribs”, the Altair technology conference, 2004.

3) Edmund Lee, Kai A. James and Joaquim R. Martin, ”Stress-Constrained Topology Optimization with Design-Dependent Loading”, Journal of Structural and Multidisciplinary Optimization, volume 46, Issue 5, November 2012.

4) AltairHyperworks user manual.